

THERMAL OPTIMIZATION OF GENSET CANOPY USING CFD

PRASHANT P. PANDAV¹, SURENDRA BARHATTE² & NITIN GOKHALE³

^{1,2}Department of Mechanical Engineering, MITCOE, Pune, Maharashtra, India

³Kirloskar Oil Engines Limited, Khadki, Pune, Maharashtra, India

ABSTRACT

In the present paper CFD based investigation has been reported in order to study the effect of change in geometry of canopy on velocity and temperature on inside of canopy. It has been observed that there is significant change temperature air flow, velocity inside the canopy. As the performance of fan is directly dependent on mass flow output. So there should be a moderate velocity and temperature profile as all these parameters are co-related. In order to predict the air flow at inlet and output and temperature in canopy, analysis is done using a 3-D CFD software and to create a general idea about a canopy, a model is created using a modelling software. Diesel engine generator sets in heavy industry plants and residential/official buildings can cause serious noise problems. Diesel generators are used very commonly in shops, offices and in industry during power shut downs. The work is aimed at optimization of noise and thermal parameters of generator set. This paper focuses on air flow through canopy. Behaviour of air inside the canopy moving over inside parts of genset. Effect of air on inside cooling of canopy and thermal analysis of canopy done in this paper. Aim of this work, thermal optimization of canopy and effect of various design on overall parameters of genset. For such work position of baffle plate, louver angle with various design simulated in 3-D CFD software. Their results discussed in this paper. By using these simulation work generator set can be examined and it gives way for investigation the different parts of genset.

KEYWORDS: Generator Set, Noise, Canopy, Optimize, Simulation

INTRODUCTION

The packaged combination of a diesel engine, an alternator and various ancillary devices such as base, canopy, control systems, louvers is referred to as a "generating set" or a "genset" for short. DG set is equipped with an engine operating as a prime mover and alternator which converts mechanical energy into electrical energy. As per CPCB norms, DG sets up to 1000 kVA should achieve noise levels of below 75dB when measured at one meter distance as per ISO 8528-10. Also temperature inside the canopy should be maintained as per CPCB. Hence it is mandatory to comply legal requirements by reducing noise pollution along with meeting end customer requirements.

Louvers are fitted to canopy. Inside canopy there is alternator, engine, fan and radiator. Louvers are attached to canopy at entry of air. Inside the canopy, air introduced from louver passes over parts of genset and absorb heat and gets heated and this air passes through radiator. Heat absorbed by coolant from engine is conveyed to the radiator and dissipated to the environment. The radiator operates with a fan, which blows hot air into the radiator, thereby promoting heat transfer from liquid coolant to air. At the time of generator running, there is vibrations and noise in surrounding area. Noise due to generators more and it creates disturbances.

METHODS FOR FAN SIMULATION

Fan simulation can be solved by various methods such as Body force model, Sliding mesh model, multiple reference frame model, moving over mesh model. Out of them MRF model discussed below.

MULTIPLE REFERENCE FRAME MODEL (MRF)

The MRF model is a steady state approximation where the fluid zone in the fan region is modeled in a rotating frame of reference and the surrounding zones are modeled in a stationary frame. In opposite to the Body Force Model the MRF-model include the geometry of the fan blades. The fan blades are modeled stationary but since the fluid domain surrounding them is in a rotating frame the pressure jump and the swirl components will be given by the presence of the fan blades as walls without the need of experimental data as an input. Even though this model clearly is an approximation due to its non time dependent approach, it can still provide realistic results for many applications.[5]

SIMULATION WORK

Simulation work for canopy CFD analysis can be done by steps of CFD calculation. These steps are important and must be followed by such sequence. To get better results for your simulation works following steps are mentioned.

Geometry

The first step is to model a canopy of genset that can be reproduced as a 3-D CAD Software engineering drawing package. Figure shows CAD design of canopy imported for simulation of canopy. In figure we shows the canopy taken for CFD analysis. This geometry was taken in the appropriate format of 3D CFD software.

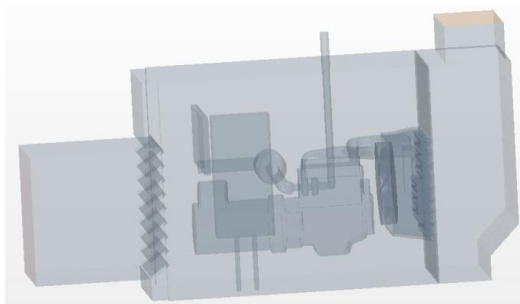


Figure 1: Geometry of Canopy

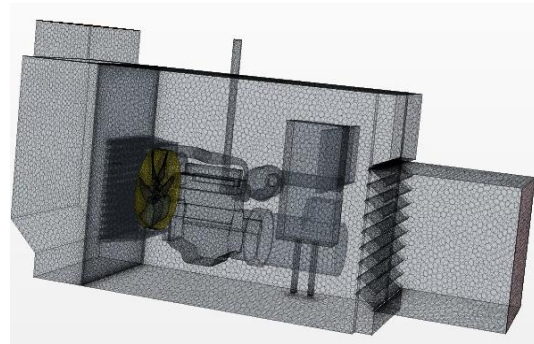


Figure 2: Meshing Model of Canopy

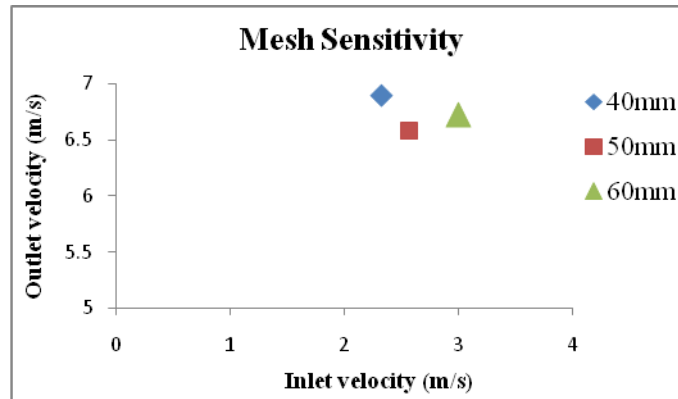
Meshing Model

A mesh is the discretized representation of the computational domain, which the physics solvers use to provide a numerical solution. The first step in generating mesh is to choose which meshing strategy to use and select the meshing models. For canopy following meshin gmodels are used in this work.

- Surface remesher : Remeshes the initial surface to provide a quality discretized mesh that is suitable for CFD.
- Polyhedral meshing : Generates a volume mesh that is composed of polyhedral shaped cells.
- Prism layer mesher : It added prismatic cell layers next to wall boundaries. The mesher projects the core mesh back to the wall boundaries to create prismatic cells.

Mesh Sensitivity

Mesh grid size is an important factor in numerical simulations because resolving of the flow motions depends significantly on the grid size to accurately describe the flows. Thus, finding or developing the right model that can satisfy the engineering solutions without the need of fine grid size (cheap computational cost) is one of the biggest challenges engineers are facing now a days.



Graph 1: Graph of Sensitivity Analysis

As the grid size gets finer, it inflicts a high computational cost because of the type of models used, such as the direct numerical simulation. In contrast, if the grid size gets coarser, this leads to use the models developed by approximations such as the Reynolds Average Navier-Stokes (RANS), which results poor description of the flow especially for the simulation of turbulent flows. Therefore, using the appropriate grid size is always crucial. From above graph we concluded that mesh size of 50mm have similar results as compared to rest of two sizes. Hence for all simulations we select mesh size of 50mm.

Boundary Conditions

Boundary conditions in the continua specify the initial field data for the simulation. For steady-state simulations, the converged solution should be independent of the initial field. However, the path to convergence, and hence the computational effort that is required to reach convergence, is affected. Therefore, choose the boundaryconditions and values judiciously, particularly when the physics is complex. Each model requires sufficient information for the primary solution data for the primary variables that are associated with the model to be set.

Boundary conditions for Canopy model are:

- Inlet: Free stream
- Outlet: Pressure outlet
- Temperature: 300K
- Flow Direction: Boundary normal
- Fan RPM: 1600 rpm

Physical Models

A physics model in CFD defines how a physical phenomenon in a continuum is represented. Essentially, physics

models define the primary variables of the simulation (such as pressure, temperature, and velocity) and what mathematical formulation is used to generate the solution. An appropriate combination of models is necessary for the complete definition of a physics model.[4]

Steady state calculations physics models

- Three dimensional
- Turbulent
- Ideal gas
- Air
- K-epsilon turbulence model
- RANS

RESULTS AND DISCUSSIONS

In this topic results obtained by the simulations work are discussed. These results are of two steady state and with 1600 rpm.

Baffle Plate above Alternator

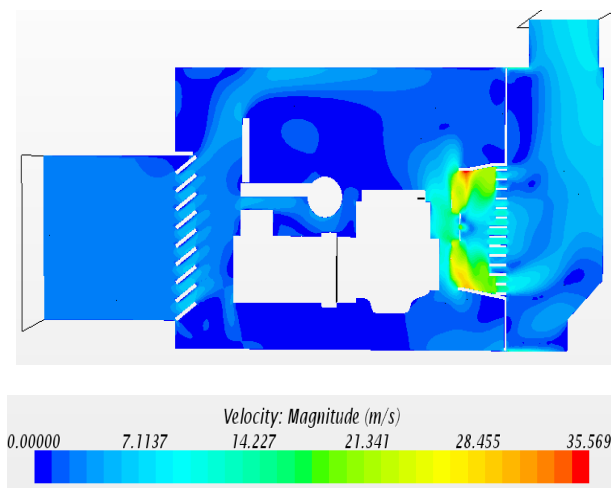


Figure 3: Velocity Contour of Canopy

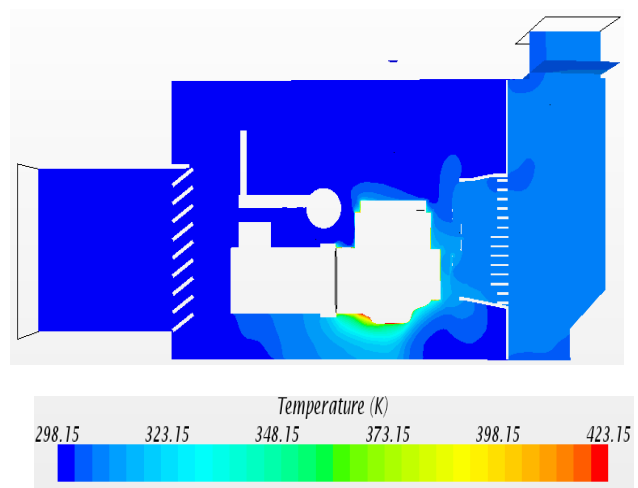


Figure 4: Temperature Contour of Canopy

Velocity and temperature contour for the original canopy as shown in figure. On these plot of velocity we clearly understand the flow inside canopy. Along with that exact readings for such flow known by these contour. At inlet velocity about 3 m/s and for the outlet 6.5 m/s. As looking towards the temperature profile of canopy, due to heat rejection there layer formed shown on surrounding of engine. This heat removed by fan fitted at front of engine. This simulation exact shows how velocity and temperature are important while running of genset.

Baffle Position at 100mm From Alternator

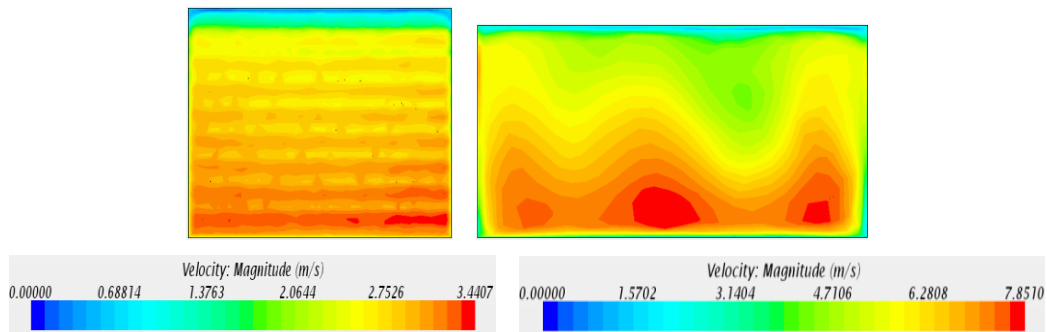


Figure 5: Velocity Contour at Inlet Louvers

Figure 6: Velocity Contour at Exit of Canopy

By change in baffle plate position we find that velocity flow at inlet of canopy improved. But when looking towards velocity at outlet, we find some region with velocity less than 4 m/s. This region directly effect of less velocity at fan inlet. It leads to rise in temperature of canopy. When comparing second position of baffle plate this is not as much suitable for genset.

Baffle Position at 20mm from Alternator Facing

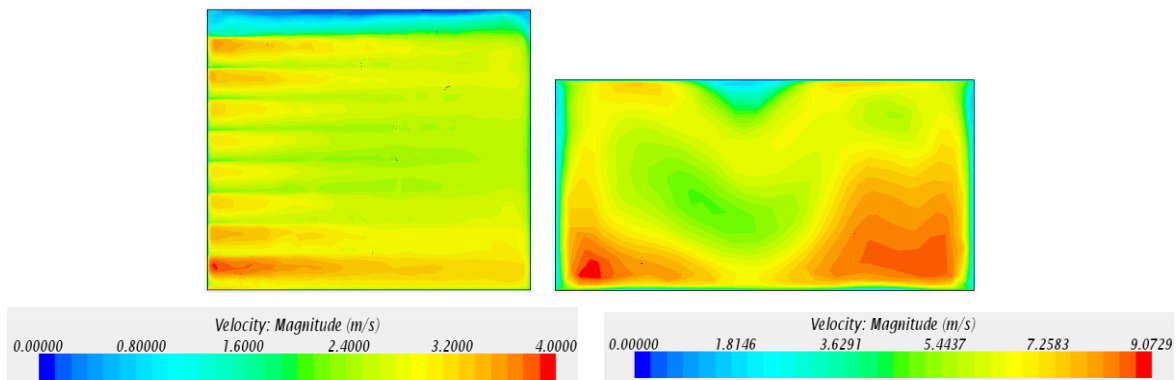


Figure 7: Velocity Contour at Inlet Louvers

Figure 8: Velocity Contour at Exit of Canopy

In this type of baffle position, velocity at inlet and exit improved by nearly 35%. This air flow reduces temperature inside canopy. Also average velocity at inlet more as compared to above position of baffle plate. Hence it is best position of baffle plate for thermal optimization of canopy.

Genset with Louver Angle 30°

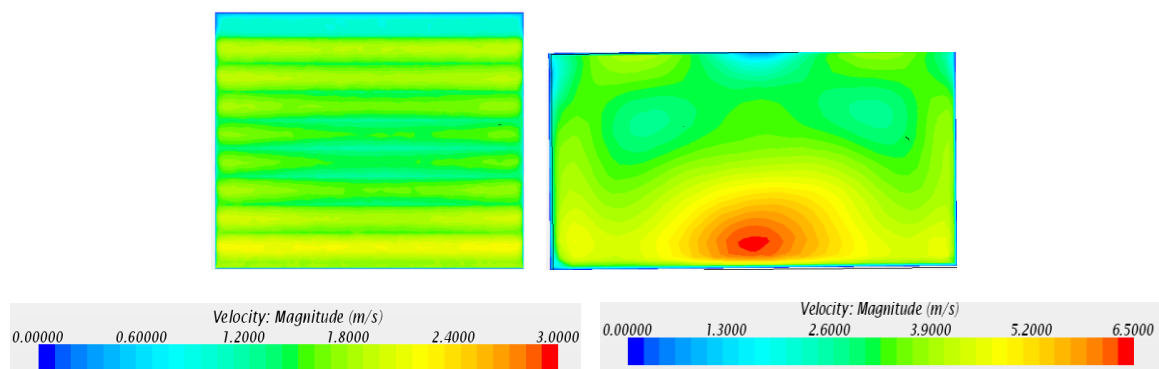


Figure 9: Velocity Contour at inlet Louvers

Figure 10: Velocity Contour at Exit of Canopy

In the above contour we see the velocity distribution is uniform across the all louvers at inlet of canopy. Bottom louvers have high velocity in previous cases but here not much difference between all the inlet louvers. For exit of canopy contour as shown represents the average velocity of 6 m/s for outlet. This is sufficient velocity for outlet to maintain temperature as per standards of canopy. So louver angle 30° best suitable for genset canopy.

Genset with Louver Angle 40°

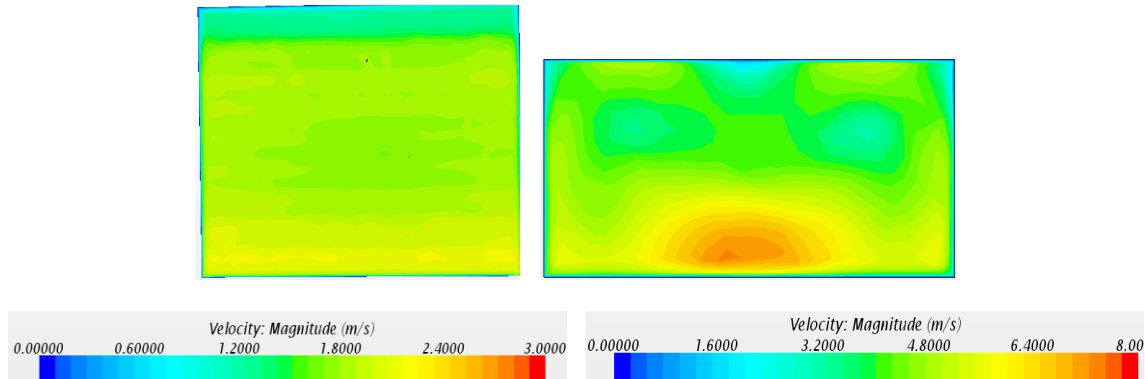


Figure 11: Velocity Contour at Inlet Louvers

Figure 12: Velocity Contour at Exit of Canopy

When simulation for inlet louvers with angle 40° conducted contour shown above for inlet and outlet. This represents that there is formation of different regions such as very minimum velocity, moderate and high. It leads to insufficient cooling of inside parts of canopy. For considering outlet plot, there is also same thing happen as different velocity regions. Temperature at outlet also rises due to such formation of zones. Also opening at inlet is more and it directly contact with atmosphere, so entry of dust particles takes place in this case. This is not suitable design of canopy for thermal optimization.

CONCLUSIONS

Analysis of different conditions of genset canopy to be conducted using CFD software. Following conclusions can be drawn from the results obtained by this work related to canopy design.

- Optimum results are obtained by changing the position of baffle plate to 20 mm from alternator. If the distance is increased to 100 mm, velocity at inlet and outlet reduces resulting in increase of temperature inside canopy due to reduced air flow. Whereas at 20mm distance from alternator, both inlet and outlet velocities are sufficient to maintain temperature inside canopy.
- For the louver angle of 40°, cooling takes place inside the canopy and velocity is higher at outlet. But due to opening of louvers, noise is not confined to the canopy only but reaches out and it leads to increased noise level.
- Louver angle 30° is best suitable for these canopy as overall temperature inside canopy and at outlet seems to be more uniform. This also follows the CPCB norms of temperature..
- Thus it is better to design a genset with 30° louver angle with baffle at 20 mm from alternator.

REFERENCES

1. System and Procedure for compliance with noise limits for Diesel Generators (Upto 1000 kVA), January 2008
2. Hyeon-Don Ju, Shi-Bok Lee, Weui-BongJeong, Byung-Hoon Lee, (2004), “*Design of an acoustic enclosure*

with duct silencers for the heavy duty diesel engine generator set", *Applied Acoustics* 65, pp 441–455

3. Srinivasa, V., S. R., and Shome, B., (2014), "*Design of Experiments Enabled CFD Approach for Optimizing Cooling Fan Performance*," SAE Technical Paper 2014-01-0658, 2014, doi:10.4271/2014-01-0658
4. Mohd Yusoff Sulaiman, Shamsul Bahari Azraai, Wan Mokhtar Wan Abdullah, (2009), "*CFD Modelling of Air Flow Distribution from a Fan*", Proceeding of International Conference on Applications and Design in Mechanical Engineering (ICADME), Batu Ferringhi, Penang, Malaysia.
5. Tobias Berg, Anna Wikstrom, (2007), "*Fan modeling for front end cooling with CFD*", Masterthesis, Lulea university of technology, Sweden.

